

Lasercamm!

The Product Realization Lab has a laser cutter that is used to cut acrylic, foamcore, paper, plywood, masonite, matteboard and a variety of thermoplastics up to 1/4 inch thick. It can also engrave, or score the surface without cutting through. The Lasercamm is connected to an HP vectra PC. It takes a DXF file and translates it into a DMC file the Lasercamm can read and then cut. It is an ideal tool for cutting intricate shapes with sharp internal corners. The maximum size the Lasercamm can cut is 36 X 21 inches. With the exception of delrin, if the material is not on the current acceptable materials list (this list appears under "material" in the Lasercamm software), you may not cut it on the Lasercamm. Please do not violate this rule - the Lasercamm is very expensive and some materials may damage the optics.

Drawing Preparation

Use a CAD package to create a drawing, and export it as a DXF file. There is no way to set coordinates, so all parts must have positive x and y coordinates. Make 0,0 the lower left corner of the drawing. It may be better to set the part origin .25 up and .25 in from 0,0 especially for big parts to ensure full cutting. **Lasercamm will abort cutting when it encounters a negative coordinate, and the software does not like grouped or blocked entities.** Ungroup all geometry before exporting from Vellum. Explode all blocks in Autocad. Each operation must be on its own layer. For example, a scored detail is separate from a cut detail. The Lasercamm will cut entities if they appear twice so be sure to clean up your drawing before exporting. If you need dimensionally accurate parts, you must compensate, in the drawing, for the width of the laser beam, and the melt/burn back. This kerf is usually between .008 and .010, so offset your part .004 to .005. You will notice that the Lasercamm will cut your part in a bizarre sequence. This is done to spread any error across the entire piece.

Text is an independent entity and can only be cut if it is composed of arcs and lines. If you are trying to cut text, you must either create the text yourself in Vellum, or find a way to convert the fonts to curves.

Transfer

If you are creating a drawing from Vellum on a shop computer you will need to do the following steps. (You may or may not need to do this if you use a different program...) Export your file out of Vellum as a DXF file. The Lasercamm reads only old format DXF files and this can be a headache when transferring files. The new DXF format inserts carriage returns where before there were only tabs. There is a small program located on the shared drive on ACME in the folder titled "LaserCamm DXF Converter." Copy it to your directory by holding down the control key and dragging the icon. You will now have a copy in your folder. Double click on "Laserpc.exe" in your directory, and follow the instructions. ("Laserpc.exe" must be in the same directory as the file you are converting.)

If you are creating your geometry on a Macintosh you will need to get your DXF file onto a DOS formatted disk.

The lasercamm software runs on a PC using DOS operating system. Put your disk into the disk drive and type **laserfl** at the prompt. This starts the Lasercamm software and uses the floppy drive, where your file is located. This can cause problems if the floppy is close to full or locked. If there is no floppy in drive a, the computer will ask abort retry, fail. The only way out of this loop is to put a DOS formatted floppy in the drive and type r.

If you are working in Autocad on the lasercamm PC, export it as a DXF file to the temporary folder or your own directory. Quit Autocad. Type **cd temp** (or whatever directory you stored your file in) then type **laserhd** at the prompt.

Using The Lasercamm Software

The software has the current directory highlighted. Press **return** to get a list of all the files there. The mouse does not work with this software so use the arrow keys to navigate the screen. Use the up and down keys to highlight your file. Press **return** to select it. The software should now list all your layers and have open spots for operation, material and thickness. Please refer to the manual if this is unclear. The Lasercamm software is a bit buggy, especially the file handling. If the PC locks up on you, remove the floppy from the drive and press the reset button and start over. Talk to a TA if you are unsure.

The possible operations are, cut, score1 through score5, and custom. Highlight the area and press return. A list will be presented to you. Use the arrow keys and select the operation. Use the right arrow to get to the material area, press **return** and use the arrow keys to find and select the material. The next thing to select is the thickness. This is tied to the material chosen. If your thickness is not there, choose the nearest one. (You can cut delrin with the acrylic setting.) It seems that changing the setting of the laser from 100% causes errors when downloading.

After you have chosen proper settings for one layer, press **F4** to process. Do what it tells you. When it asks if you should send to the Lasercamm, answer yes if it is the only layer you have. If you have multiple score and cut layers, select no and process the other layers as explained above. **Ignore any focus settings the software gives you.**

After you have processed all your layers, you can combine them. Press **F6**, then select the DMC files from the list using the arrow keys and the enter key. It is best to do all score processes before cutting. The combined list is done in order - first in first out. After you have your combined list, press **F4** to combine. It will ask you for a target file. Enter a file name not longer than 8 characters, no spaces. (i.e. if your DXF file is coolpart.dxf, you could use coolpart again) To send your file press **F9** to send. Be sure the Lasercamm is on before sending.

Using Lasercamm The Machine

Insert the key and turn it. wait for the Lasercamm to initialize before sending a file. After sending the file, set the focus based on the list on the side of the machine. The focus is independent of material type. **Ignore any focus numbers the software gives you.**

Each turn of the barrel is .5. Improper focus can cause the part not to get cut all the way through, as well as burns back excessively. Get a TA to check this if you are unsure. Check the display. Make sure the proper file is being cut as well as the proper thickness and material. The file name listed is that of the DXF file that the work group came from.

First cover the bed of the Lasercamm with paper. This is important - by covering the bed strong suction is created where the laser cuts through material and paper. This sucks all the smoke down and away from the optics. Smoke will damage the optics as well as smell terrible, so if your part is smoking too much, fix it. The laser is probably not cutting all the way through your material - check the settings again, or contact a TA.

Place material in Lasercamm and close cover. Switch into jog mode and use the four arrow keys to position the head over your workpiece. When you are satisfied, turn off the jog mode. There is no provision for accurate positioning of the head. This stopped position is the 0,0 coordinate of your drawing.

Press **start** (green) button. The fan should automatically turn on after the start button is pressed. (The fan may be turned on manually by pressing the FAN button on the front panel.) If there is an emergency press **E stop** (big red button), but NOT IF THERE IS A FIRE! If you press the big red button and there is a fire below your piece, the laser will stop above the flames. Hit the **stop** button instead and the laser head will move back to the origin. **Stop** will reset the program to the beginning. If you want to pause the work, you can press the **pause** (top black rocker switch) button and cutting will stop at the end of the current command. Lifting the Lasercamm cover will also stop the laser from cutting. **Be sure to wait at least 1 minute before pressing start (green) button again. Failure to do so can send the cutting head crashing into the stops and ruin the alignment of the laser beam.** It has been found that stopping in the middle can cause the machine not to restart properly. If this happens, press in the big red button, wait one minute and then rotate the big red button out. Wait for the machine to initialize. Your file should still be there ready to cut. If this does not remedy this situation, report the problem to a TA.

Please refer to the user manual if something in this handout doesn't make sense. And don't forget that your TAs are a valuable source of info. Thanks for reading and happy cutting.

revision history

rev 0-3	5/95	John Wadsworth	original text, revisions
rev 4	5/97	Bryan Cooperrider	revisions and updates